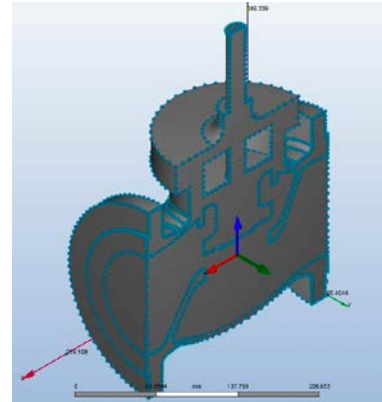


Lesson: Unsteady flow setup

In this lesson, you are introduced to unsteady flow and how it can affect other considerations in the design such as the thermal conditions of the model.

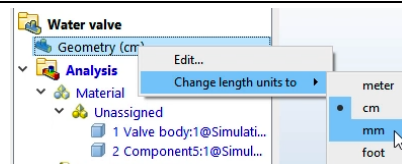
Learning Objectives:

- Understand the simulation workflow.
- Set up boundary conditions for the study.
- Create a transient load case for the study.

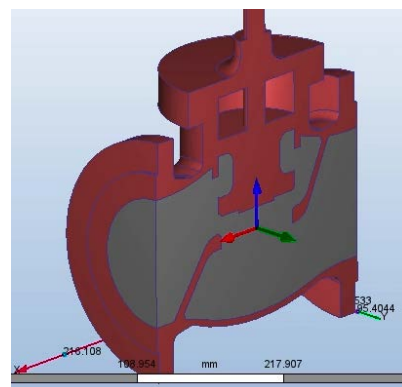


The completed exercise

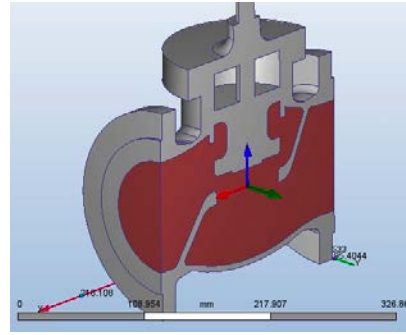
1. In Autodesk CFD 2019, open the file Unsteady Flow.cfdst and note that the model has the unified valve body well as the water volume body. In the Browser, right click on Geometry then Change the length units and then click mm.



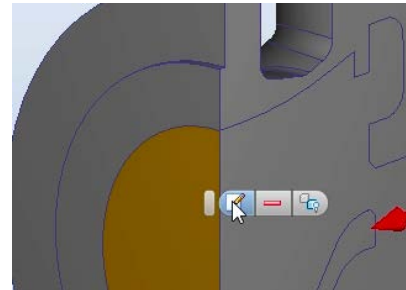
2. In the Browser, select the Valve body then right click and select Edit. In the Materials control panel, change the type to Solid then change Name to Steel. Click Apply.



3. Repeat step 2 in order to edit Component5. Change the Type to Fluid and change the Name to Air. Click Apply.



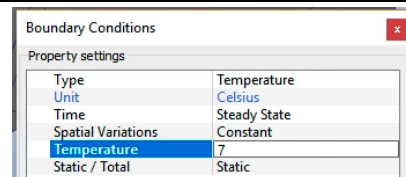
4. Click Setup>Boundary Conditions. Click on the semi-circular face of the inlet body then Edit. Change the Unit to m/s, change the time to Transient, and change the Time Curve to Piecewise Linear.



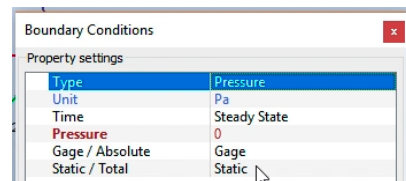
5. In the Time Curve control panel enter the values shown in the picture here and click plot to see a graphical representation of the values you just entered. Close the plot window. Click OK then click Apply.

	Value	Time (sec)
1	.2	.2
2	.5	.4
3	1	.6
4	1.2	.8
5	1.5	1
6	1.5	5

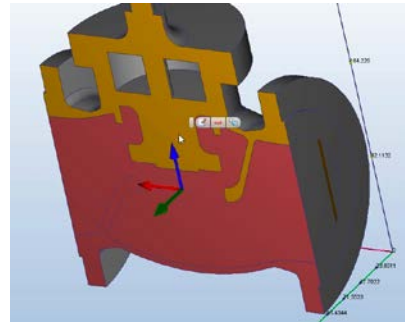
6. Select the same face and Edit. Change the Type to Temperature and change the Temperature to 7 degrees C. Click Apply.



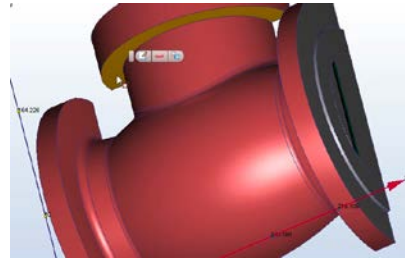
7. Rotate the model so you can see the outlet then select the semi-circular face of the fluid body and Edit. Change the Type to Pressure and set the value to 0. Click Apply.



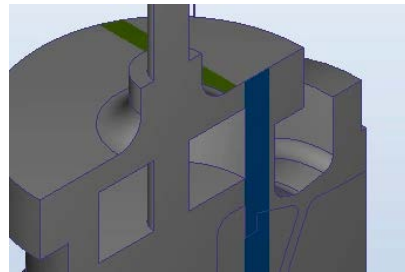
8. Select the faces that make up the plane where the water and valve was bisected. Click Edit and change the type to Slip/Symmetry. Click Apply.



9. Select all the external faces of the valve not including the flange faces which will mate to other pipes and click Edit. Change the Type to Film Coefficient and set its value to 10. Click Apply.



10. Note the five different colored icons in the lower left corner, each representing a different boundary condition. The faces on the model will have a strip that corresponds to the color of the boundary condition it contains.



11. Click Setup>Mesh Sizing. Note the various options then click Autosize.

